COMPUTATIONAL FLUID FLOW ANALYSIS OF A SIDE MIRROR FOR A PASSENGER CAR

PRABAGAR S/O MURUKESAVAN

Report submitted in partial fulfillment of the requirements
for the award of Bachelor of Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

JUNE 2012

ABSTRACT

Today, reducing the carbon dioxide emissions is vital. The car industry has a responsibility to reduce the fuel consumption and will thereby reduce carbon dioxide emissions. One of the main questions in the automotive industry is how to go about this. One possibility is to change the propulsion system. Another option is to reduce the aerodynamic drag of the car; the topic of this thesis. The drag is of great importance when it comes to velocities over 60 km/h. There are many parts of the car that contribute to drag. One such part is the side-view mirrors. The mirrors increase the total amount of drag by 2-7 percent. The mirror plays a major role in drag contribution for the entire car and therefore mirror optimization is considered very important. Mirror optimization is not an easy task due to uncertainties in the CFD simulations of a few drag counts which makes it impossible to trust all findings. In order to find a good mirror design, a combination of wind tunnel testing in full scale, and CFD simulations is necessary. Mirror design optimization shows great potential. This thesis describes the evaluation of aerodynamic flow effects of a side mirror towards a passenger car based on the side view using ANSYS Fluent CFD simulation software. The parameters that are found in this research are pressure coefficient, total pressure, drag coefficient and lift coefficient. The pressure coefficient of the side mirror designs is evaluated to analyze the unsteady forces that cause fluctuations to mirror surface and image blurring. The fluctuation also causes drag forces that increase the overall drag coefficient, resulting in higher fuel consumption and emission. There are 3 types of model tested in this research. The model is tested in simulation using the speeds of 16.67m/s (60km/h), 25m/s (90km/h) and 33.33m/s (120km/h). The models are then compared using their drag coefficient and lift coefficient. The results indicate that the half-sphere design shows the most effective design with less pressure coefficient which causes fluctuation and has low drag and lift coefficient.

ABSTRAK

Hari ini, mengurangkan pengeluaran karbon dioksida adalah penting. Industri kereta mempunyai tanggungjawab untuk mengurangkan penggunaan bahan api dan dengan itu akan mengurangkan pengeluaran karbon dioksida. Salah satu persoalan utama dalam industri automotif adalah bagaimana untuk mendalami penambahbaikan dalam perkara ini. Satu kemungkinan ialah dengan menukar sistem tujahan. Satu lagi pilihan adalah untuk mengurangkan seretan aerodinamik kereta iaitu topik tesis ini. Heret adalah amat penting apabila halaju melebihi 60 km/h. Terdapat banyak bahagian kereta yang menyumbang dalam seretan. Salah satu bahagian adalah cermin pandangan sisi. Cermin ini meningkatkan jumlah seretan sebanyak 2 hingga 7 peratus. Cermin ini memainkan peranan yang penting dalam sumbangan seretan untuk keseluruhan kereta dan oleh itu pengoptimuman cermin sisi dianggap sangat penting. Pengoptimuman cermin sisi bukan satu tugas yang mudah kerana ketidaktentuan dalam simulasi CFD menjadikannya mustahil untuk mempercayai semua penemuan. Untuk mencari reka bentuk cermin yang baik, kombinasi ujian terowong angin dalam skala penuh, dan simulasi CFD adalah perlu. Pengoptimuman reka bentuk cermin menunjukkan potensi yang besar. Tesis ini menerangkan penilaian kesan aliran aerodinamik cermin sisi ke arah kereta penumpang berdasarkan pandangan sisi dengan menggunakan simulasi perisian ANSYS Fluent CFD. Parameter yang terdapat dalam kajian ini adalah pekali tekanan, jumlah tekanan, pekali seretan dan pekali daya angkat. Pekali tekanan dinilai mengikut reka bentuk cermin dan tekanan akan menyebabkan getaran permukaan cermin dan kekaburan imej. Getaran juga menyebabkan daya seretan yang meningkatkan pekali seretan keseluruhan, mengakibatkan penggunaan bahan api yang lebih tinggi dan pelepasan. Terdapat 3 jenis model yang diuji dalam kajian ini. Model ini diuji dalam simulasi menggunakan kelajuan 16.67m/s (60km/h), 25m/s (90km/h) dan 33.33m/s (120km/h). Semua model ini kemudian dibandingkan dengan menggunakan pekali seretan dan pekali daya angkat. Keputusan menunjukkan bahawa reka bentuk separuh sfera merupakan reka bentuk yang paling berkesan dengan pekali tekanan yang kurang yang menyebabkan getaran dan mempunyai seretan rendah dan pekali daya angkat rendah.

TABLE OF CONTENTS

LIST OF TABLES

LIST OF FIGURES

LIST OF SYMBOLS

%	Percentage
ρ	Density
km/h	Kilometer per hour
m/s	Meter per second
m	Meter
V	Velocity
A	Frontal Area
L	Length
kg	Kilogram
m^3	Meter cube
kPa	Kilo Pascal
P_{static}	Static Pressure
$P_{dynamic}$	Dynamic Pressure
P_{atm}	Atmospheric Pressure
P_{Total}	Total Pressure
C_D	Drag Coefficient
C_L	Lift Coefficient
F_D	Drag Force
F_L	Lift Force
k-ε	K-Epsilon

LIST OF ABBREVIATIONS

2D	Two Dimensional
3D	Three Dimensional
ANSYS	Analysis System
CAA	Computational Aero Acoustics
CFD	Computational Fluid Dynamics
DNS	Direct Numerical Simulations
FEM	Finite Element Method
FVM	Finite Volume Method
FYP	Final Year Project
LES	Large Eddy Simulations
RANS	Reynolds Averaged Navier-Stokes

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 BACKGROUND

Reducing fuel consumption, and therefore reducing the carbon dioxide emissions, is one of the most important goals in today's car industry. One way this can be achieved is by reducing the engine size, using an electric motor with a combustion engine, reducing the weight of the car and reducing the aerodynamic drag of the car. The latter is of great importance when it comes to velocity over 60km/h. Above this velocity, the aerodynamic resistance is higher than the rolling resistance (Versteeg, H.K., 2007). Streamlined body design in a passenger car helps reducing the aerodynamic drag and eventually improves the engine mileage. On the contrary, accessories attached to the body skin of a car cause the unfavorable aerodynamic examples. In order to obtain the rear sight, unfortunately the mirror does not pay only the aerodynamic penalty which increases body form drag, but also causes the acoustic noise thus causes mirror fluctuations to the cabin crews.

While the aerodynamic body styling of the passenger car has been upgraded with a lot of efforts, the defects caused by important accessory such as the side view mirror have been ignored. The main stream meets a side flow which has the flow direction tangent to the windshield surface near the A-pillar. And a conical vortex sheet is generated along the pillar and merges into the mainstream. Therefore, very complicate flow pattern appears by combining these flow patterns near the driver side window. Moreover, since the side mirror is mounted on the driver door near hinge, the wake flow behind this obstacle become much complicated. (D. Gillespie, 2000)

1.2 PROBLEM STATEMENT

Reducing fuel consumption, and therefore reducing the carbon dioxide emissions, is one of the most important goals in today's car industry. One way this can be achieved is by reducing the aerodynamic drag of the car. It is a great importance when it comes to velocities over 60km/h. Above this velocity, the aerodynamic resistance is higher than the rolling resistance. Streamlined body design in a passenger car helps reducing the aerodynamic drag and eventually improves the engine mileage. (H.K. Versteeg, 2007).

On the contrary, accessories attached to the body skin of a car cause the unfavorable aerodynamic examples. In order to obtain the rear sight, unfortunately the mirror does not pay only the aerodynamic penalty which increases body form drag, but also causes the acoustic noise and the mirror fluctuations to the cabin crews.

The main function of rear mirror is to provide drivers the rear view of a vehicle. Rear view side mirror contributes to aerodynamic drag, noise and vibration. The aerodynamic drag can be reduced to lower the fuel consumption rate.

1.3 OBJECTIVES

The objectives of the project are as follows:

i. To study the effects of aerodynamic flow towards a passenger car side mirror based on the side view (x-y axis)
ii. To compare different types of side mirror designs using CFD
iii. The best design which has better flow characteristics is chosen as a base model.

1.4 SCOPES OF STUDY

The scopes of the project are as follows:

i. Designing a full scale of the side mirror structure attached to a passenger car model.
ii. Design 3 new designs of side mirror for comparisons.
iii. Run analysis using a CFD application which is ANSYS Fluent to obtain results by using different speed (km/h) parameter.
iv. Compare the obtained computational results with experiment method results by F. Alam Et.Al., 2007.

1.5 SIGNIFICANT OF STUDY

Computational Fluid Dynamics (CFD) is done to save cost and time of running high cost experiments using experimental wind tunnel. The research could reduce the consumption of fuel and produce less harmfull emissions to the atmosphere during the usage of automotives. From this research, it will help to increase the importance of usage of computational results to analysis a certain experiment with high cost and long duration.

1.6 STRUCTURE OF REPORT

This report is consists of five chapters. The first chapter is introduction about the research study. It includes the background of study, problem statement, objectives, scopes, significant of the study and the structure of report.

Next chapter focuses on the literature review based on the previous aerodynamic experiments done. Besides that, this chapter includes the fundamental of aerodynamic, the factors that influence aerodynamic and parameters associated with the aerodynamic effect assessment. So, this chapter has major influences to increase better understanding on this research study and is very helpful to design the methodology of study.

Chapter 3 describes the methodology of the procedures of this research which was done. The flow chart of this research is presented with potential arising issues with the preventive action plans. The methods and procedures are described in general. The designing of wind tunnel, passenger car and side mirror are explained. Besides that, the details of used simulation settings and boundary conditions are briefly described. Other than that, the data analysis method is also explained at the end of this chapter.

The results and discussions are presented in next chapter. The generated data of pressure coefficient, total pressure, drag coefficient and lift coefficient are presented in form of graphics for qualitative discussion. The contours of flow are also discussed to understand the best design characteristics. The correlation between the experiment result and simulation result is also discussed. Finally, the outcome of this study was compared with previous wind tunnel experiment done by researches.

The final chapter of this thesis consists of conclusion of this research study. The overall conclusion with finalized details is discussed with some recommendation for improvements proposed.

CHAPTER 2

LITERATURE REVIEW

2.1 COMPUTATIONAL FLUID DYNAMICS (CFD)

Computational fluid dynamics (CFD) is a computer simulation that analyzes systems for fluid flows, heat transfer, and phenomena such as chemical reactions. The rapid development of computational power and CFD technique, the field of Computational Aero Acoustics (CAA) becomes more and more relevant to the industrial applications, and this method has been applied in the area of the aerospace industry , meteorology (weather prediction), and external environment of buildings (wind loads and ventilation) commonly. CFD has many advantages over experiment-based approaches, such as reduction of lead times and costs of new designs, study systems under hazardous conditions, systems that are impossible to study with controlled experiments and, the unlimited level of detail of the results.

There are also problems with CFD. The physics are complex and the result from CFD is only as good as the operator and the physics embedded. With today's computer power, there is a limitation of grid fineness and the choice of solving approach (DNS, LES and turbulence model). This can result in errors, such as numerical diffusion, false diffusion and wrongly predicted flow separations. The operator must then decide if the result is significant. While presently, CFD is no substitute for experimentation, it is a very helpful and powerful tool for problem solving.

Concerning the comfort of driver, more and more attention is paid to noise in the car development process. Flow induced noise, generated by additional device at the vehicle body, i.e. Side mirrors, antennas or spoilers are especially important. (H. K. Versteeg, 2007)

When working with CFD a number of different steps are followed. These steps are illustrated in figure 2.1.

Figure 2.1: The CFD process flow

The first step is to create geometry (with CAD). This is often already done by other departments or done by scanning a model. The geometry cannot have any holes, it has to be airtight, and unnecessary things in the CAD model that do not affect the flow has to be removed to save computer power. This is called CAD cleanup. The next step is to generate a mesh and this is often done automatically by a meshing program. Then the flow is simulated by a solver. After the simulation is ready, it is time for post processing. Post processing involves getting drag and lift data, and analyzing the flow.

2.1.1 CFD solving approaches

There are many types of solving approach, one of that is the Direct Numerical Simulations (DNS). It solves the Navier-Stokes equation numerically. This will resolve all the different turbulent scales. The solution will be transient and requires a very fine mesh with sufficiently small time steps. Due to the extreme grid size and number of time steps required for a simulation at high Reynolds number, this approach is not today possible (lack of computer power).

Then there is the Reynolds-Averaged Navier-Stokes (RANS) method. It gives an approximate time-averaged solution to the Naiver-Stokes equation and focuses on the mean flow properties. The fluctuating velocity field, also called Reynolds stress, has to be modeled. But this turbulence model cannot solve all turbulence scales.

The last approach used in this experiment is the Large Eddy Simulations (LES) it computes the larger eddies in a time-dependent simulation while the universal behavior of the smaller eddies can be captured with a model. LES uses a spatial filtering operation to separate the larger and the smaller eddies. (D. Gillespie, 2000)

2.2 HISTORY OF AUTOMOTIVE AERODYNAMICS TECHNOLOGY

Aerodynamics and vehicle technology have merged only very slowly. A synthesis of the two has been successful only after several tries. This is surprising since in the neighboring disciplines of traffic technology, naval architecture, and aeronautics the cooperation with fluid mechanics turned out to be very fruitful. Of course, the designers of ships and airplanes were in a better position. They found their originals in nature from fish and birds. From these natural shapes they took many essential features. The automobile had no such originals. Hence its designers tried to borrow shapes from ships and airplanes, which must have appeared progressive to them. Very soon this turned out to be the wrong approach. Only when it broke away from these improper originals did aerodynamics make a breakthrough in the automobile.

Another reason for the early repeated failures of aerodynamics with vehicles is that it started far too early. The first automobiles were pretty slow. On the bad roads of those days streamlined bodies would have looked ridiculous. Protecting driver and passengers from wind, mud and rain could be accomplished very well with the traditional design of horse-drawn carriages. Later the prejudice that streamlined bodies were something for odd persons overrode the need for making use of the benefits of aerodynamics for economical reasons. A brief overview of the history of vehicle aerodynamics is summarized in Figure 2.2.

During the first two of the total four periods, aerodynamic development was done by individuals, most of them coming from outside the car industry. They tried to carry over basic principles of aircraft aerodynamics to cars. Later, during the remaining two periods, the discipline of vehicle aerodynamics was taken over by the car companies and was integrated into product development. Since then, teams, not individual inventors, have been responsible for aerodynamics.

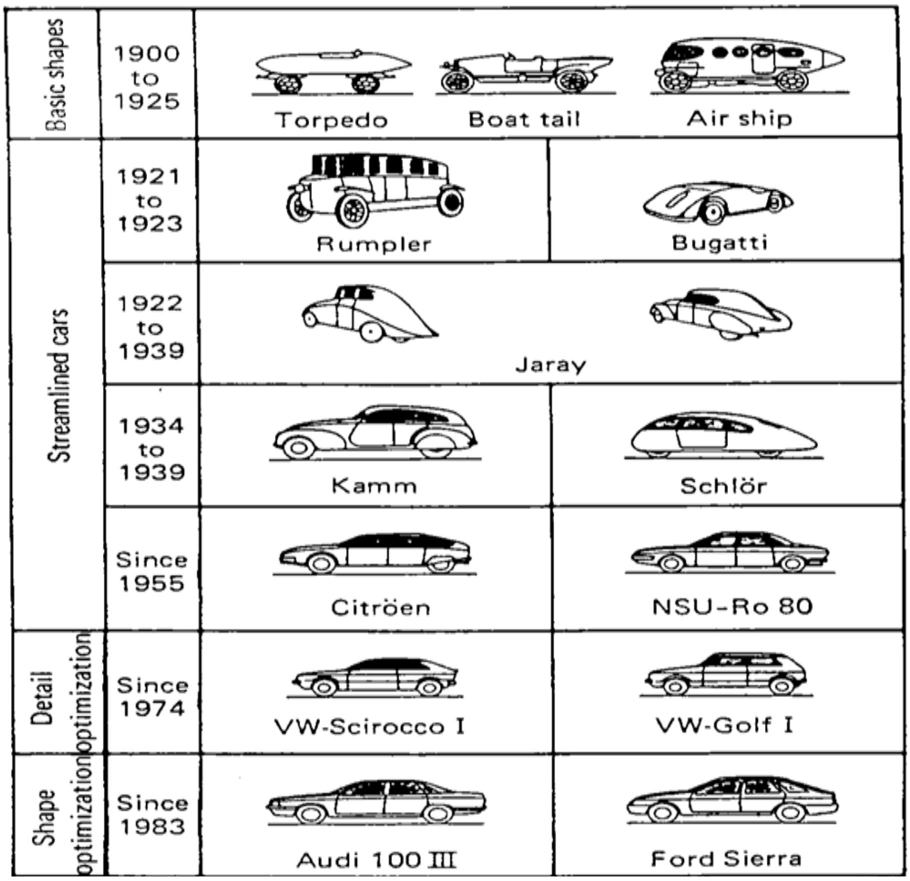

Figure 2.2: History of vehicle dynamic in passenger car

Source: D. Gillespie (2000)

The first automobile to be developed according to the aerodynamic principles was a torpedo-shaped vehicle that had given it a low drag coefficient but the exposed driver and out of body wheels must have certainly disturbed its good flow properties. However they ignored the fact that the body was close to the ground in comparison to aircrafts and underwater ships flown in a medium that encloses the body. In a car like this, the ground along with the free-standing wheels and the exposed undercarriage causes disturbed flow. As the years pass the studies on aerodynamic effects on cars increase and the designs are being developed to accommodate for the increasing needs and for economic reasons. The wheels developed to be designed within the body, lowering as a result the aerodynamic drag and produce a more gentle flow. The tail was for many years long and oddly shaped to maintain attached the streamline. The automobiles became developed even more with smooth bodies, integrated fenders and headlamps enclosed in the body. The designers had achieved a shape of a car that differed from the traditional horse drawn carriages. They had certainly succeeded in building cars with low drag coefficient (D. Gillespie, 2000)

2.3 AUTOMOTIVE AERODYNAMICS

Aerodynamics of cars became more and more important with the increase of their velocity. In the beginning of the 20th century, the shape of vehicles was adopted from the field of aviation and ships. Cars had an aerodynamic shape but their velocity was very low, mainly due to the quality of the roads. Aerodynamics is the branch of dynamics that deals with the motion of air and other gaseous fluids and with the forces acting on bodies in motion relative to such fluids. Automotive aerodynamics is the study of the aerodynamics of road vehicles. The main concerns of automotive aerodynamics are reducing drag, reducing wind noise, minimizing noise emission and preventing undesired lift forces at high speeds. For some classes of racing vehicles, it may also be important to produce desirable downwards aerodynamic forces to improve traction and thus cornering abilities (D. Gillespie, 2000). An aerodynamic automobile will integrate the wheel and lights in its shape to have a small surface. It will be streamlined, for example it does not have sharp edges crossing the wind stream above the windshield and will feature a sort of tail called a fastback or Kammback or lift back. It will have a flat and smooth floor to support the venturi or diffuser effect and produce desirable downwards aerodynamic forces. The air that rams into the engine bay, is used for cooling, combustion, and for passengers, then reaccelerated by a nozzle and then ejected under the floor. Most everyday things are either caused by aerodynamic effects or in general obey the aerodynamic laws. For aerodynamic bodies are simplifies procedure may then be devised for the evaluation of the aerodynamic loads.

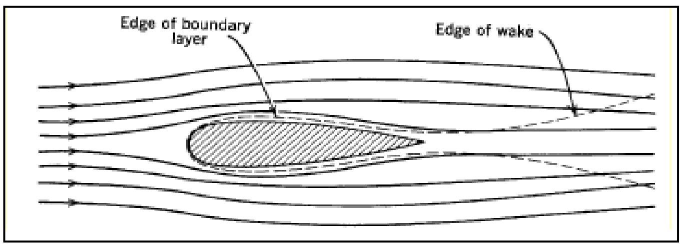

Figure 2.3: Aerodynamic of bluff bodies

Source: D. Gillespie (2000)

A car driven in a road is affected by aerodynamic forces created. The aerodynamics of such cars is of vital importance. They affect the cars stability and handling. They influence both performance and safety.

2.4 AERODYNAMICS DRAG

The force on an object that resists its motion through a fluid is called drag. When the fluid is a gas like air, it is called aerodynamic drag (or air resistance). When the fluid is a liquid like water it is called hydrodynamic drag. Drag is a complicated phenomena and explaining it from a theory based entirely on fundamental principles is exceptionally difficult.

Fluids are characterized by their ability to flow. In semi-technical language, a fluid is any material that can't resist a shear force for any appreciable length of time. This makes them hard to hold but easy to pour, stir, mix, and spread. As a result, fluids have no definite shape but take on the shape of their container. Fluids are unusual in that they yield their space relatively easy for other material things at least when compared to solids. Fluids may not be solid, but they are most certainly material. The essential property of being material is to have both mass and volume. Material things resist changes in their velocity and no two material things may occupy the same space at the same time. The portion of the drag force that is due to the inertia of the fluid is the resistance to change that the fluid has to be pushed aside so that something else can occupy its space is called the pressure drag. (A. Cengal, 2006)

$$C_d = \frac{F_d}{\frac{1}{2}\rho v^2 A} \tag{2.1}$$

Based on Eq. (2.1), where F_d is the Drag force, C_D is the Drag coefficient, ρ is the fluid density, A is the frontal area and v is the Velocity.

2.5 DRAG FORCE

The drag equation for an object moving through a fluid is as followed

$$F_d = \frac{1}{2}\rho\upsilon^2 C_d A \tag{2.2}$$

Based on Eq. (2.2), where F_d is the force of the drag, ρ is the density, υ is the velocity, C_d is the drag coefficient and A is the reference area. The most important variables are the reference area (frontal area of the car) and the drag coefficient. By reducing these, the aerodynamic drag will be reduced which will lead to lower fuel consumption rate.

2.6 AERODYNAMICS LIFT

Lift is normally of little importance in passenger cars as their speed is usually too low to produce much lift. It was noticed early on that something strange happened at high speeds: the car seemed to be lifting off the ground. The lift can be serious, particularly in racing cars. It has a serious effect on the control and handling of the car. (A. Cengal, 2006)

Lift occurs because the airflow over the top of a car is faster than across the bottom. This occurs to some degree in all cars. As the speed increases, the pressure decreases, according to Bernoulli’s theorem. The top of the car therefore has a lower pressure than the bottom, and the result is a lifting force. The amount of lift generated by an object depends on a number of factors, including the density of the air, the velocity between the object and the air, the viscosity and compressibility of the air, the surface area over which the air flows, the shape of the body, and the body's inclination to the flow, also called the angle of attack.

$$C_L = F_L / (1/2\ \rho V^2 A) \tag{2.3}$$

Based on Eq. (2.3), where F_L is the lift force, C_L is the Lift coefficient, ρ is the fluid density and A is the frontal area, while v is the velocity

2.7 AERODYNAMIC PRESSURE

The gross flow over the body of a vehicle is governed by the relationship between velocity and pressure expressed in Bernoulli's Equation. Bernoulli's Equation assumes incompressible flow which is reasonable for automotive aerodynamics. (A. Cengal, 2006)

$$P_{static} + P_{dynamic} = P_{total} \tag{2.4}$$

$$Ps + \frac{1}{2}\rho V^2 = Pt \tag{2.5}$$

Based on Eq. (2.4) and Eq. (2.5), where ρ is the density of air in kg/m^3 and V is the velocity of air (relative to the car) in m/s

In the equation above, the sum of the forces brings in the pressure affect acting on the incremental area of the body of fluid. The static plus the dynamic pressure of the air will be constant (Pt) as it approaches the vehicle. At the distance from the vehicle the static pressure is simply the ambient, or barometric, pressure (P_{atm}). The dynamic pressure is produced by the relative velocity, which is constant for all streamlines approaching the vehicle. As the flow approaches the vehicle, the streamlines split, some going above the vehicle and others below. By inference, one streamline must go straight to the body and stagnate (impinging on the bumper of the vehicle). At that point the relative velocity has gone to the zero. This will make the static pressure observed at that point on the vehicle. Figure 2.4 below shows flow over a cylinder that it affects is most same to the vehicle's coefficient. (D. Gillespie, 2000)

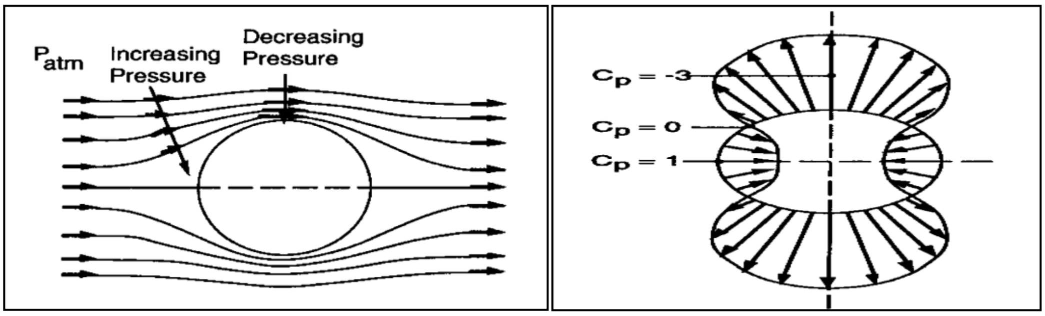

Figure 2.4: Pressure and velocity gradients in the air flow over the body coefficient

Source: D. Gillespie, (2000)